Use of NS3D Code in the Design of HPT Blade Cooling Air Supply Systems

M. Balland, S. Coulon, F. Monchois
Heat Transfer and Secondary Flow Department
Bat. 40, Niv. 7
SNECMA, Villaroche
77550 Moissy Cramayel, France

ABSTRACT

Competition in aeronautical propulsion imposes new challenges in terms of maintenance costs and product availability. Within the HP turbine design process, aero-thermal engineers are given only a short time in which to propose adequate solutions for HPT blade cooling air supply systems. The industrialization of the SAVIRAT chain at SNECMA MOTEURS offers opportunities for applying Navier-Stokes calculations which are now compatible with the quality and time constraints of a project design phase.

One clear way to improve the life time of HPT disks is to remove hooks between the forward sealing plate and the HPT disk itself. In addition to its main mechanical function, this device also optimizes the relative speed of the HPT blade cooling air before it enters the slot bottoms.

This document details the contribution made by Navier-Stokes 3D analyses during the HPT disk design phase, in the event of a simple contact, without hooks, between the forward flange and the disk rim. A fluid and solid wall 3D mesh represents the HPT cooling air supply system, from the forward sealing plate exit holes to the entry of the cooling air in the blade root. Calculations were carried out under adiabatic conditions.

Models have been created for several configurations. These have led us to draw the following conclusions:
- where a “no hook” device is used, analysis reveals a vortex phenomenon due to bailing,
- in the case of a conventional solution with hooks, the vortex phenomenon disappears and the pressure distribution in the slot bottoms is homogenous,
- fitting blade roots with axial air deflectors restores air supply quality in slot bottoms.

To conclude, the state of industrialization of the SAVIRAT chain at SNECMA MOTEURS allows aero-thermal engineers to use 2D or 3D Navier-Stokes calculation supports for design work. Some applications may have a significant impact on the optimization of components, in terms of efficiency or maximizing life time. Secondary flows of turbine rotors are one of the most interesting fields of application.

1. INTRODUCTION

Today, improvement in engine performance means, for the engine designer, increasingly stringent specifications in the sizing of HP turbines. Over the last thirty years, turbine inlet temperature values – a decisive factor in the overall output of the engine – have doubled, and today reach temperatures of around 1800 °C. Furthermore, the specified life times of HP turbine components have remained the same, or are indeed greater. In parallel to this, peak operational temperature of new materials is changing less rapidly (see fig.1), meaning that the improvement of materials is no longer the only response when faced with these new requirements.

Paper presented at the RTO AVT Symposium on “Advanced Flow Management: Part B – Heat Transfer and Cooling in Propulsion and Power Systems”, held in Loen, Norway, 7-11 May 2001, and published in RTO-MP-069(I).

Report Documentation Page

Form Approved
OMB No. 0704-0188

1. REPORT DATE	2. REPORT TYPE	3. DATES COVERED
00 MAR 2003	**N/A**	**-**

4. TITLE AND SUBTITLE	
Use of NS3D Code in the Design of HPT Blade Cooling Air Supply Systems	5a. CONTRACT NUMBER
	5b. GRANT NUMBER
	5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S)	5d. PROJECT NUMBER
	5e. TASK NUMBER
	5f. WORK UNIT NUMBER
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) **NATO Research and Technology Organisation BP 25, 7 Rue Ancelle, F-92201 Neuilly-Sue-Seine Cedex, France**	8. PERFORMING ORGANIZATION REPORT NUMBER
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)	10. SPONSOR/MONITOR'S ACRONYM(S)
	11. SPONSOR/MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release, distribution unlimited

13. SUPPLEMENTARY NOTES
Also see ADM001490, presented at RTO Applied Vehicle Technology Panel (AVT)Symposium held in Leon, Norway on 7-11 May 2001, The original document contains color images.

14. ABSTRACT

15. SUBJECT TERMS

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT	b. ABSTRACT	c. THIS PAGE			
unclassified	**unclassified**	**unclassified**	**UU**	**12**	

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std Z39-18

In this context, the internal ventilation system of the HP turbine plays an essential role. Firstly, it is responsible for cooling the rotor parts, such as the disk (which is particularly massive) and the sealing flanges, in order to limit transient gradients and to ensure a longer life time. But above all, it supplies the blade cooling system with air, a function which must provide both a minimum pressure level, in order to avoid any risk of air from the air-flow reentering the blade assembly, and the lowest possible air temperature.

For the designer, the impact of these increasingly stringent specifications means that the sizing of the HP turbine ventilation system has to be even more precise, with the aim of achieving the greatest possible optimization of convective exchanges, limiting purges on flowpath, and meeting the supply conditions of blades. It is possible, using 1D tools, to quickly evaluate very different architectures during the definition phase. These tools are also widely used in precise design, as they remain well adapted for 90% of the configurations studied.

However, the experience gained using these tools does not cover certain more innovative ventilation systems. It is therefore essential to have CFD resources that make it possible to better understand the nature of the flows, to identify the phenomena that cause load losses, and to evaluate these configurations. With this in mind, SNECMA has introduced the SAVIRAT chain, to make 3D Navier-Stokes calculations of internal flows of turbo-machines. The chain is built around the MSD solver developed by ONERA, which is particularly adapted to subsonic internal flows and which is sufficiently robust to perform calculations in cavities with complex-geometries.

The document presents a concrete example of how the SAVIRAT chain is used in engine design. Following a presentation of the different design stages, the first section of the document deals with describing the chain itself, along with how it is integrated into the designer’s environment. A second section provides an example that clearly shows the gains that can be obtained by using these tools on a specific HP turbine ventilation system configuration.

2. THE DESIGN PROCESS AND NS3D TOOLS

2.1 The design process

There are two major stages in the internal turbine ventilation system design process (fig. 2):

The first stage, known as the definition phase, is aimed at defining and evaluating several possible blade air supply system configurations, then choosing the most suitable architecture. Here, the ventilation tools used are mainly 1D. These codes are based on a semi-empirical resolution of conservation of mass and energy equations in stationary and rotating cavities. A number of correlations of tests on standard restrictions are integrated at this stage (stationary and rotating orifices, labyrinth seals, etc.). These allow the most suitable architecture to be quickly determined. The 2D or 3D Navier Stokes analysis is used for systems of which there is little or no prior experience. This makes it possible for designers to evaluate performance of more promising solutions. Here, equivalent digital simulations are used for comparing the different solutions.

The second stage is the design phase, involving an in-depth study into the configuration chosen above. Various thermo-mechanical iterations make it possible for the final geometry to be reached. During this phase, the 1D ventilation tools are coupled with heat transfer solver. Thermal data is then used to determine the maximum mechanical stress level of the parts, as well as their life time. Several thermo-mechanical iterations are necessary in order for the geometry of the parts to be optimized. 2D or 3D Navier-Stokes calculations are performed on certain sections of the system, in order to reduce the uncertainties associated with the 1D model, for complex cavities or flows involving a large number of 3D components. These studies account for less than 10% of the overall volume carried out in the design phase, but have a great impact on the optimization level of the components, and therefore on their performance and life time.

2.2 Description and integration of the SAVIRAT chain

At the heart of this tool chain lies the MSD (Mathilda Saphir Diamant) solver, developed in collaboration with ONERA. This is used to solve Navier-Stokes equations in the compressible domain, while stationary, for perfect gas mixtures. The resolution technique is based on a method using finite volumes in a structured, multi-domain mesh. Several turbulence models are available, but the standard model that is used is a K-L model with two transport equations. For calculations in areas close to walls, the solver shall automatically propose wall laws where a rough mesh is used. This code is able to process a wide range of flows, but is particularly well suited to low-Mach configurations. Although slower than the transonic codes used in external aerodynamics at SNECMA, it does have a major strength in that it enables practically all types of internal engine cavities to be processed with a high level of stability.

In order to be able to use the MSD code in design operations, substantial efforts have been made to integrate user-friendly interfaces for processing standard data formats currently used at SNECMA MOTEURS. This suite, referred to as the SAVIRAT chain, is made up of several modules:

- Pre-processing of the SAVIRAT chain (fig. 3) is performed by a suite of applications from ICEM Technologies that share a single interface. Two tools, CATIA and DDN, are used to process the design office CAD files. As the design office also uses CATIA, raw data can be processed directly, thus saving a considerable amount of time.
- The second major stage is meshing. Thanks to the ICEM/HEXA software, the user is able to create structured meshes of complex geometries. The volume is broken down into multiple topological domains whose sides are then projected onto the geometry. The advantage of this technique is that meshing created in this way is highly adaptable to close geometries. It is also possible to select only a part of the model for meshing. Finally, an interface developed with the MSD solver enables boundary conditions to be applied in the mesher. The user may then use verification and analysis tools, such as ICEM/LEO, to quickly evaluate the quality of its meshing.
- Finally, the post-processing tools used are the market-standard visualizors - IRIS Explorer and ENSIGHT 7 – which essentially enable iso-value fields or lines of current to be analyzed. In parallel to this, a set of internally developed tools is used for the analysis and monitoring during the iterations of physical parameters. This final function also makes it possible to obtain the best possible setting of the convergence parameters during calculation.

In addition to this, validating the SAVIRAT chain on the basis of test configurations representing engine cavities allows the designer to maintain a constant level of quality. Each change in MSD code version number is therefore validated by calculations made for the engine systems under experimental testing. Furthermore, SNECMA MOTEURS is an active participant in European programs such as Brite-Euram ICASGT (Internal Cooling Air System of Gas Turbines), which perform experiments on “typical” cavities (rotor-rotor, cavities under compressor guide vane assemblies, etc.) and compare these to Navier-Stokes calculations.

As well as its software resources, SNECMA also has high-capacity processing resources. Notably, the user has a 16-processor Fujitsu VPP300 vectorial computer at its disposal. With the aid of internally developed compilation tools, it is also possible to carry out calculations on SGI R12000 workstations. Finally, additional processing capacity is available on CEA parallel computers.

3. ROLE OF THE HP TURBINE INTERNAL VENTILATION SYSTEM

The HP turbine's rotor ventilation system has several roles. Firstly, it must ensure that the pressure, air flow and temperature levels at the blades internal cavity entrances are sufficient for optimal operation. It must also ensure cooling for the rotor parts in order to limit transient gradients and to meet the life time objectives. The Balance of axial thrusts also have an impact on the architecture of the system, as these particularly affect the radii of the labyrinth seals. Finally, leaks that occur in the flow path must be minimized to ensure the performance of the turbine is not adversely affected.

The "typical" system shown in this document (fig. 4) is made up of a bleeding from the combustor discharge at an air temperature close to that of the compressor outlet. This bleeding constitutes the system's cold source. Air is then fed to the fixed pre-swirl nozzle, where it is greatly accelerated and rotated in order to lower its total relative temperature. It then crosses over the revolving holes of the sealing flanges which role is to limit inner leaks to the flow path. The air becomes compressed and moves up to the rim, from where it is supplied to the slot bottoms of the disk and the turbine blade systems.

On many turbine architectures, the upstream flange is held on to the turbine disk by a system of crenellated hooks called claws (fig. 4b). This hook system avoids any risk of top-end opening as a result of the combined effects of centrifugal and heat loads, and maintains the lowest possible level of leakage. It also has a very important impact on ventilation: as the air moves up to the slot, its rotation speed falls as its kinetic momentis maintained. The air is then fed into the hooks, which drive it back up to rotor speed. In this case, losses at the slot inlet are minimized, and flow into the slot is clean and predictable.

However, the inconvenience of the claws is that they generate large areas of mechanical stress which limit the rotor's life time. In addition to this, they are expensive to manufacture. In the ventilation configuration presented here, these hooks have therefore been removed. In terms of ventilation, air is no longer driven at rotor speed by the lateral surfaces of the claws, and its driving coefficient $K=\omega air/\omega rotor$ continues to decrease as the radius increases. The situation at the slot inlet is therefore very different as, this time, the air is moving at a tangential speed lower than that of the rotor, and no longer enters the slot axially. This configuration is out of current experience and is difficult to capture with a 1D code. The use of a Navier-Stokes code is all the more justified here in that a precise knowledge of load losses has a direct impact on the supply pressure of the blade assembly.

4. IMPLEMENTING A NS3D MODEL

4.1 Description of the models used

Two models were used in this study. The first model was made using a reference geometry (fig. 5) similar to the real geometry, which allowed a comparative study to be made on how the claws influence the pressure levels in the slots. This principle is adapted to the definition phase when the geometry of the parts is not fixed in detail. The second model shown (fig. 11) was created at the end of the design phase using the final geometry, with the addition of a pressure correction system in the slot bottom.

For these two models, the meshing area was an angular segment corresponding to a slot and the channels of one blade. The reference model included rotating holes and the rotor-stator pre-swirl nozzle cavity. The choice of these interfaces is closely linked to the boundary conditions, and it is therefore important to extend the model to a well-known area that is sufficiently far away from the area under study, in order to limit the effects of these on the results. Analysis of the first calculations led us to extend the inlet surface only to the disk-flange cavity, in order to limit the meshing to the area under study.

There are four types of boundary conditions:
- repetitiveness conditions that enable an entire ring to be simulated without it being necessary to make a complete model of the ring, thus constituting a large saving in calculation time,
- wall conditions,
- inlet pressure conditions (Pt, Tt, Kent = Wair/Wrotor),.
- static pressure conditions for the outlet of the blade vane assembly channels.

Inlet and outlet conditions are based on 1D models data.

4.2 Reference model

The results of the reference geometry revealed a pressure dip in the middle of the slot (fig. 7a, 7b) which at this point reached more than 1.2 bars. This dip was identical in Ps and Ptr, which would tend to indicate that speeds in this area are very low. The radial cross-section shows that the pressure dip corresponds to the entrance of an internal blade cavity. Analyses in relative Mach and driving coefficient revealed the turbulent structure of the flow (fig. 8). Air is evacuated via the pressure side where it is greatly accelerated (M>0.3), and then re-circulates towards the wall on the suction side rotating around the center of the slot. An area of low pressure is therefore created by the effects of viscosity and inertia of the fluid.

In order to verify that the origin of the phenomena is indeed linked to the pre-rotation of the air in front of the slot, the basic model was influenced so as to simulate the presence of the claws between the flange and the disk. Making a model showing the way in which air is channeled by the slits of the claws is complex, and requires the meshing to be completely regenerated. Only the lateral surfaces of the claws were therefore simulated, as these are the only ones that are involved in channeling the air. They were simulated using wall conditions applied directly to a mesh plane (fig. 6). This technique makes it possible to exert influences within time constraints compatible with the design cycle. The result given in chart 5 shows that air does indeed arrive at a rotation speed close to that of the rotor in front of the slot. The resulting field of pressure (fig. 9) is much more homogenous and maximum differences are reduced by 75%. Air enters axially into the slot and its drive remains close to 1. As the channeling section is not contracted by a bailing effect, the relative mach is 3 times less (M~0.1) than on the model with no claws (fig. 10).

These results show that, on one hand, the pressure dip of the first configuration is indeed the result of air rotation in the slot and that, on the other hand, the turbulent movement is generated by the difference in tangential speed between the air and the rotor at the inlet of the slot.

In order for the architecture without claws to be viable from an aerothermal point of view, it is necessary to limit the vortex phenomenon, as this is damaging to the blade cooling system. The decreasing of air flow caused by the pressure deficit may lead to a rise in the average temperature of the blade and may halve its potential life time. Moreover, the possibility of this pressure deficit spreading along the supplies of the blade’s leading or trailing edge, according to the flight scenario, cannot be ruled out. This would lead to a lower margin of re-introduction of hot gases, and would cause localized burns. During the course of the mission, rotation speeds and pressure ratios are subject to change and, consequently, relative air speed may vary and significantly modify the bailing angle at the slot inlet (Driving Coefficient K=0.8 at full throttle, 0.85 at idle). An immediate solution would be to raise the blade supply pressure in order to compensate for the vortex phenomenon. But this pressure surplus would result in a significant increase in inner leaks and a degradation of the turbine efficiency (-0.2% SFC).

The following chapter describes the system set up to limit the vortex phenomenon, and the use a NS3D calculation to analyze its effectiveness.

4.3 Adaptation to a real geometry

The principle of the solution chosen for limiting the pressure dip consists of guiding the air in the slot and separating it into several different flows. This allows the turbulent movement to be limited and to homogenize the air flow. The air is guided by the vertical sides of a ‘U’-shaped metal sheet placed under the blade root, thus forming two fins. The first model was used to determine the length and position of these fins in the slot. Wall conditions were imposed on the meshing of the slot in order to simulate different configurations. This technique made it possible for several solutions to be quickly evaluated without it being necessary to modify the reference meshing.

The geometry finally chosen is shown in figure 13. It takes into consideration constraints of manufacturing and mechanical resistance, and its main difference in relation to the geometrical influences performed is the thickness of the rail. In order to validate this configuration, a new mesh was created retaining 90% of the previous model’s topological section. Only the modeling of the deflector required re-cutting to define its vertical sides in the slot. In order to evaluate the configuration without fins, all that is necessary is to redefine the solid volume of the fins as a fluid volume. The other modifications made to the basic model are as follows:

- number of blade supply channels reduced from 7 to 4,
- lengthening and reduction of the slot area,
- number of blades increased of 20-> modification of the angular area under study.

All of these modifications are carried out on the topological section upon which the initial mesh is based. All that is then necessary is to project the sides of the areas adapted in this way onto the new geometry. This technique enables 70% of the initial model to be reused, thus making it possible for the user to create a catalog of “typical” topological models that may be reused on demand. The final meshing is shown in fig. 12.

Finally, the size of the model is reduced to the area around the slot, thus concentrating the meshing – while retaining the same number of points – on the area of interest. The furthest upstream boundary of the model is a surface where pressure and temperatures were found, when examined, to be homogenous with the reference model, in the air gap between the turbine disk and the flange. The boundary conditions applied are updated directly in the mesher on the most severe temperature operating point for the blade assemblies.

The results of this research are shown in figures 14 & 15. The configuration without the deflector confirms the presence of a vortex generating a dip in pressure of 1.4 bar at the center of the slot, even though the pressure and temperature conditions are different from the first calculation scenario. The addition of the rail system makes it possible to reduce this pressure dip by nearly 1 bar, thus achieving a gain of 71 % in relation to the configuration without fins (figure 14b). Moreover, the location of the pressure dip is now towards the bottom of the slot, whereas in the initial configuration it supplied the 3^{rd} channel. Analysis of the mach values clearly reveals the guidance provided by the rail system. A double bailing effect is obtained at the slot inlet, allowing a part of the air flow that had previously circulated in the pressure side to be extracted, to directly supply the channels of the blade vane assembly. The air flow that is held in the pressure side flows into the chicane formed by the deflector at the bottom of the slot. The role of the small fin on the suction side is to ensure that the flow does not circulate back onto the suction side, recreating a rotating flow that may have an adverse effect on pressure levels. Even if this reversal is a source of load loss, it nevertheless allows air speed to be reduced ($M=0.22$-> $M<0.1$) and avoids a vortex phenomenon being formed. On the other hand, the air that flows underneath the channels causes an increase in pressure at the end of the reversal due to a mixing effect. Even if a residual vortex still remains, its scale is greatly scaled down which largely reduce the pressure deficit (figure 15b) ; Its center is situated on the line at position $K=1$ (driving at rotor speed). The length of the suction side fin was established in such a way that this residual pressure dip is situated between the 3^{rd} and 4^{th} channel where it interferes as little as possible with the supply of the blade internal cavities.

The NS3D study therefore confirms the merit of installing a deflector at the blade root. The reason that the calculations were not made for every flight characteristic is that the configuration chosen proved itself effective in the most severe conditions in regards to blade cooling. These analyses are all the more reliable as they are based on differences between a model without deflector and the same model fitted with a deflector. On the other hand, the vortex phenomenon was observed in two slot geometries that were very different in terms of section and length, and which featured bleed channels of different sizes and number.

The understanding of the flow inside the internal system made it possible to find the solution best adapted to the problem caused by the removal of claws. The deflector proposed is a lightweight part that costs little to manufacture and which generates low fatigue stresses on the blade root.

4.4 Lead times

The time it took to carry out the research presented in this document can be broken down as follows:

- pre-processing,
- meshing,
- calculation,
- post-processing and analysis of the results.

The following table shows the estimated number of hours that were necessary to complete the studies for these different tasks.

Estimated time \ Model		Model 1	Model 1 with claws	Model 2 with deflectors	Model 2 without deflectors
Pre-processing (CAD + Meshing)		8 days	0.5 days (adaptation of Model 1)	4 days (adaptation of Model 1)	0.5 days (Adaptation of Model 2)
Calculation (Fujitsu VPP 300)	Total No. of hours for 1 calculation	165 hours	105 hours	74 hours	69 hours
Post processing / analysis of the results		10 days	7 days	5 days	5 days

(model 1: reference model; model 2: real geometry model)

Table 1: calculation times for the models shown.

Care must be taken when examining the calculation times as these depend on a number of factors such as the computer, the calculation quota attributed to the user or the occupation of the machine at the time of submission. Moreover, the convergence of the code itself has a direct effect on the duration of the calculation. Further upstream, the complexity of the geometry studied and the quality of the meshing are of paramount importance. On the other hand, the choice of convergence parameters may enable precious time to be saved by increasing or decreasing the time needed to stabilize the calculation.

Studying these times shows us that, in terms of time, the use of NS3D tools may be realistically envisaged within a design cycle.

5. CONCLUSION

This document has provided a presentation of the NS3D analysis resources available at SNECMA MOTEURS for the purpose of internal aerodynamics. The use of these tools was extended from the domain of research and process engineering, and was integrated into the engine design cycle. The research carried out along these lines shows the suitability of such tools for shaping the internal ventilation system. These assets may be summarized as follows;

- firstly, the possibility for the designer to better understand the nature of the flows in configurations of which we have little experience, and to highlight the phenomena that may cause pressure deficit, with a view to putting forward solutions for improvements. This is what occurred for the vortex phenomenon in the slot bottom which would not have been brought to light with 1D tools only. Here, the air supply to the turbine blade is directly affected. A pressure deficit in relation to the specifications therefore has a direct consequence on the life time of the component, and so on the performance and reliability of the turbine.

- secondly, the ability of the chain of SAVIRAT tools to be integrated in and adapted to the design process. Modifying close geometries can be done with a degree of flexibility that allows the designer to quickly visualize influences and to check the validity of the solutions put forward. The way in which the tools can be adapted to the complex geometries of the internal cavities and the robustness of the MSD solver make it possible to guarantee the time it will take to carry out operations, which is important when planning the design cycle.

While it is true today that the industrialization phase of the NS3D tools has been a success at SNECMA MOTEURS, efforts must nevertheless be continued in order to maintain, or raise, the quality of the SAVIRAT chain. With this in mind, different operations are under consideration:

- continual improvement of turbulence and transition models is performed in collaboration with ONERA. The aim is to achieve a better analysis of load losses but also of the heat exchanges in the internal cavities,
- reduction in calculation time by the optimization and parallelization of the MSD code. This objective also involves increasing the processing resources at SNECMA,
- addition of types of boundary conditions such as permeability curves,
- creation of a catalogue of meshing models for the pre-processing phase,
- improvement of post-processing tools to give opportunity for engineers to make the most of calculations results

When completed, these improvements will make it possible for designers to be offered NS3D analysis solutions that are compatible with development constraints and highly complementary to 1D tools.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge the technical direction of SNECMA Moteurs for the authorization to publish the present paper. They wish also to thanks ONERA for their contribution to CFD development. Moreover, It must emphasized that CFD tools presented in this paper have been carried out by many co-workers in technical direction

REFERENCES

1 Duboué, Liamis, Paté. : Recent Advances in Aerothermal Turbine Design and Analysis (2000) 36th AIAA Joint Propulsion – paper 3355

2 Dutoya, D.&Errera M.P (1191): le code MATHILDA. Modèles Physiques, réseau de calcul et méthode numérique. Rapport technique ONERA n°42/3473EN

3 Montenay A.Paté L,Duboué J.M (2000) : Conjugate Heat transfer Analysis of an Engine Internal Cavity. ASME paper 2000-GT-282

4 JM. Owen, E.D Bilimoria, Heat transfer in rotating cylindrical cavity, J. Fluid Mech.

Paper Number: 28

Name of Discusser: J. Rabiega, Poznan University of Technology, Poland

Question:
1 The time of solving the problem was CPU time.
2 “Abaqus” is not a flow code, it is for heat conduction at least.
3 How expensive was the 5 week license for the compretational code used?

Answer:
1 The time of five weeks include the mesh, the calculation and past-processing for both configurations.

2 We just use Abaqus as a conduction solver. For internal flow, MSD is our Navier Stokes 3D CFD solver.
3 As we contribute to in terms of the development of the MSD Code, there is no licence needed.

Name of Discusser: E. Oktay- Roketsan Turkey

Question:
How do you calculate heat transfer coefficient?

Answer:
We make a calculation with adiabatic walls to obtain a reference temperature field.
A second calculation with imposed temperature on the walls give the heat flux
of which we extract a heat transfer coefficient field . The data fields are defined in every cell of the mesh.
We can do iterations between CFD calculation and conduct on solver to achieve the right wall temperature field.

Name of Discusser: Weyer, DLR Cologne

Question:
Did the numerical study help you to analyse and understand the origin of the vortex?

Answer:
Yes, absolutely. The Navier Stokes 3-D study allow us to discover the vortex phenomena in the slot and it’s origin. We wouldn’t have found it with 1-D or even Navier Stokes 2-D tools.
It’s a good example of the advantages of this type of analysis.

Name of Discusser: S. Bock, MTU Aero Engines Munich

Question:
What kind of turbulence model did you use?

Answer:
We use a k-l turbulence model with combined a wall law.

Name of Discussor: B. Simon, MTU Aero Engines Munich

Question:
Did you use your results without validation and how you would perform validation in this case.

Answer:
We use the results of this study without validation on this case. But, first, the MSD Code have been composed and validated for several years on realistic engine cavities. Second, we always work on relative comparison basis to evaluate a system and we make two calculations.
To my opinion it's the best way to evaluate a new system with good confidence.

For the second point, we are going to make a test engine and pressure and temperature will be measured in the slot. Besides, several blade will not be equipped with the anti-vortex system. So we could see the difference on the metal temperature of the blade with thermal paint analysis.

This page has been deliberately left blank

Page intentionnellement blanche